Araştırma Makalesi **BAUN Fen Bil. Enst. Dergisi, 25(1), 107-121, (2023)**

DOI:10.25092/baunfbed. 971558 ***J. BAUN Inst. Sci. Technol., 25(1), 107-121, (2023)***

Geometric improvement for two bridge pier types to reduce local scour

Volkan KİRİÇÇİ[1]*, Ahmet Ozan ÇELİK[1]

[1]*Eskisehir Technical University Faculty of Engineering, Department of Civil Engineering, Iki Eylul Campus, Eskisehir,Turkeu*

Geliş Tarihi (Received Date): 14.07.2021
Kabul Tarihi (Accepted Date): 12.11.2022

Abstract

Local scour around the bridge piers play a critical role in collapse and heavy damages on bridges. Many researchers have studied local scour problem. However, majority of the studies only focused on examination of the complex flow structure and offering new pier geometry designs without any structural restrictions, which ignore construction difficulties. This study attempts to investigate the effect of geometric enhancements on commonly used conventional pier types to reduce scour related flow parameters acting on riverbed. A right triangular implementation that wraps the reference shapes were accordingly applied as non-load bearing curtain walls. Triangular implementation has been parametrized by its leg lengths and successive numerical analyses were conducted to reveal the better option among different modifications.

Performance of the modified piers were compared in terms of pressure and the bed shear stresses acting on riverbed representing the two important flow forces that are responsible for triggering and developing scour. The results show that low height values with high width values of conical implementations perform better in terms of reducing maximum pressure and shear stress acting on riverbed.

***Keywords**: CFD, bridge pier, scour, modeling.*

Lokal oyulmayı azaltmak amacıyla iki köprü ayağı tipi için geometrik iyileştirme

Öz

Köprü ayakları çevresinde oluşan lokal oyulma, köprü çökmelerinde ve köprülerde meydana gelen ağır hasarlarda kritik bir rol oynamaktadır. Birçok araştırmacı lokal

---

*Volkan KİRİÇÇİ, vkiricci@eskisehir.edu.tr, http://orcid.org/0000-0001-8856-2021
Ahmet Ozan ÇELİK, aocelik@eskisehir.edu.tr, http://orcid.org/0000-0002-8770-2894

oyulma sorununu incelemiştir. Ancak, çalışmaların çoğunluğu sadece karmaşık akım davranışının incelenmesine odaklanmış, herhangi bir yapısal kısıta bağlı olmayan ve inşa zorluklarını göz ardı eden yeni köprü ayağı tasarımları önerilmiştir. Bu çalışmada, sıklıkla kullanılan konvansiyonel köprü ayağı tiplerine uygulanan geometrik iyileştirmelerin, nehir yatağında oyulmaya neden olan akım parametreleri üzerindeki etkilerinin incelenmesi amaçlanmıştır. Referans geometrilerin etrafını dolaşan, herhangi bir taşıyıcı özelliği olmayan dik üçgen biçimindeki eklentiler uygulanmıştır. Söz konusu dik üçgen biçimindeki eklentiler uzunlukları cinsinden parametrize edilmiş ve uygulanan farklı modifikasyonlar arasından en iyi seçeneğin ortaya çıkarılması için ardışık nümerik çözümler gerçekleştirilmiştir.

Değişiklik yapılan geometrilerin performansı, oyulmayı tetikleyen ve gelişiminden sorumlu olan iki önemli akım parametresi olan nehir yatağına etkiyen basınç ve kayma gerilmesi değerleri üzerinden değerlendirilmiştir. Sonuçlar, küçük yükseklik ve büyük genişlik değerine sahip konik eklentilerin, nehir yatağına etkiyen maksimum basınç ve kayma gerilmelerini düşürmesi bakımından daha etkili olduğunu göstermiştir

***Anahtar kelimeler**: HAD, köprü ayağı, oyulma, modelleme.*

1. Introduction

The design of bridges those are crossing rivers are special structures that require multidisciplinary efforts. Evaluating the structural safety of a bridge under only static and earthquake loads are insufficient without hydraulic considerations. Studies show that the majority of the collapses and heavy damages of bridges occur due to riverbed scour around the piers which is also called as local scour or clear water scour [1-3]. Bridge piers being obstacles to the river flow changes the natural riverbed and threatens the structural stability. Main factors effecting local scour can be listed as flow characteristics, pier geometry and soil characteristics. Complex flow behavior around the bridge piers differ according to the upstream flow conditions and shape of the pier. Kinetic energy of the flow acting on upstream part of the bridge pier converts into stagnation pressure. As a result, vertical velocity gradient results in high pressure gradient from top to bottom on the nose of the pier. This high-pressure gradient leads to downward water flow which is responsible for scour formation at the frontal area of the pier. In addition, flow separation due to adverse pressure gradient at the pier surface and the interaction of approaching streamwise flow and the downward water jet produce complex vortex structures which travel through to downstream. Overall, these flow mechanisms are important driving factors in erosion of streambed material around piers.

A considerable amount of literature based on different experimental approaches and empirical formulations have been published on investigation of flow characteristics and determination of key factors effecting local scour parameters [4-9]. Generalized assumptions of experimental studies with certain limitations and lack of widely accepted formulations reveal the importance and necessity of numerical studies. Computational Fluid Dynamics (CFD) method based on modeling and analysis of complex fluid dynamics problems using numerical techniques has provided important contributions to scour modelling. Richardson and Panchang [10] used a 3D hydrodynamic model to determine the maximum scour depth around pier, results of which were in good agreement with the experiments and reported inconsistencies in the results of existing formulations. Salaheldin et al. [11] investigated flow field around a cylindrical bridge pier numerically using k-ε turbulence model. Despite the general bias in literature about the weakness of k-ε model when modeling such complex flow conditions, accurate velocity profiles were obtained in this study. However, in terms of inconsistency between calculated bed shear stresses, accuracy and reliability of the numerical model studies were also discussed. In another numerical study using k-ε turbulence model conducted by Zhu and Liu [12] the results from the CFD model were in good agreement with that of experiments. Xiong et al. [13] analyzed parametrically the flow around different bridge piers and scour development using standard k-ε turbulence model and a dynamic mesh approach. Accuracy of the used model in determining the scour profiles is emphasized. Nasr-Allah et al. [14] analyzed scour mechanism around the bridge piers on sand soil for different contraction ratios using experiments and numerical models latter with k-ε turbulence closure. It was shown that the scour depth is increasing with the increase in contraction. In the same vein, Li and Tao [15] performed a transient CFD analysis coupled with Discrete Element Method (DEM) to simulate dynamic scour mechanism around an oblong-shaped pier. Results helped resolve the relationship between the drag force and the entrainment of microscopic stream bed particles.

In addition to investigation and analysis of flow and scour mechanism around the bridge piers, various studies have attempted to reduce and prevent the scouring by changing pier geometry or using soil protection methods. The study by Cheiw [16] proposed slot and collar application as an alternative to the rip-rap layer which are commonly used to protect the river bed and reduce scouring around the bridge piers. Results showed that the slot width of a quarter of the pier radius reduce the depth of scour by up to 20%. It was also emphasized that the reduction ratio can be increased by using collars together with the slots. Kumar et al. [17] experimentally examined the effects of different slot and collar configurations on scour showing that the slot lengths which reach the river bed and the wide collars located close to the river bed have significant impact on reducing scour. Another study conducted by Zarrati et al. [18] was aimed to reduce and control the scour around the rectangular bridge piers using different collar configurations and types. Fotherby [19] offered mats, grout bags, footings and tetrapod as four alternatives to rip-rap layer. Required application details of these protection methods are discussed and results of experiments show that the tetrapod has higher stability than rip-rap option. Pasha et al. [20] compared different bridge pier shapes to reveal the relation between scouring and pier shapes. According to the results, a pier with elliptic cross section causes 15% lower scour depth than a pier with diamond cross section and 10% lower scouring depth than bridge pier with circular cross section. Moreover, Vijayasree et al. [21] investigated flow field around rectangular, oblong, trapezoidal, triangular and lenticular pier shapes. Experimental results revealed that the shape of the piers plays an important role in scour mechanism. Trapezoidal-nosed, triangular-nosed and lenticular shapes were observed to be more effective options due to smaller frontal area where high pressure zones occur. Yagci et al. [22] performed a series of experiments to examine clear water scour around a finite array of cylinders with different configurations and solid volume fractions compared to a solid cylinder of equal circumambient diameter. While the arrays with higher solid volume fractions causes similar scouring behavior with the solid cylinder, array with the lower volume fractions causes more distinctive scour patterns. Some of the array configurations used in this study resulted in 22% lower scouring depth and 27% smaller scouring volume than the solid cylinder.

Overall, all of these studies highlight the need for new approaches or enhancing the presented suggestions to prevent clear water scour problem around the piers. Particularly, majority of the studies remain narrow in offering new pier geometry designs without any structural restrictions. The specific aim of this study is to numerically examine the simple but effective geometric modifications for commonly used conventional bridge pier types. While doing so, the study uses the numerical techniques effectively together with a parametric approach to draw conclusion on the performance of practicable new pier geometries.

2. Method

CFD modelling is one of the effective and practical ways of assessing different geometric modifications for bridge piers. In this study, two commonly used pier shapes with constant cross section, namely the circular and oblong types were analyzed as the reference cases. A series of modifications were applied on these 3D geometries to compare near bed stresses. Stagnation pressure at the frontal surface of the pier and the flow separation are the two critical flow mechanisms that dominate the clear water scour. The modifications, in an attempt to manipulate the flow and these two mechanisms, consist of sloped wall round the circular and oblong reference geometry creating cone shaped pier skirts. Dimensions of cone shaped modifications are parametrized by the lengths of the legs of right triangle spinning around the base geometry (Figure 1).

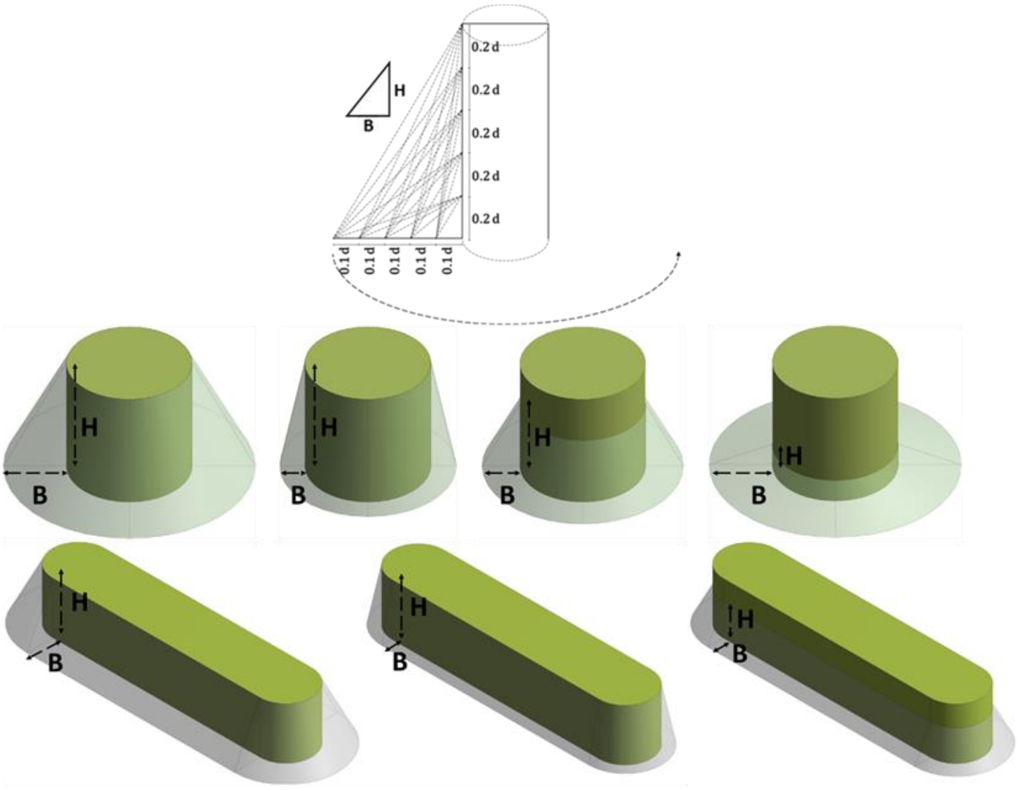


Figure 1. Schematic view and 3D demonstration examples of implemented modifications parametrized by the B/H ratio (figure is not scaled)

Performance of the modified piers is assessed in the framework of pressure and wall shear stress values acting on riverbed as the direct outcomes of the stagnation and separation mechanisms. Several studies comparing the piers with different cross-sections proved that streamlined shapes for piers lead to significant reduction in scour. Li and Tao [23] conducted 2D and 3D CFD analysis to examine the effect of curvature of the pier cross section on scour potential. Shape of the streamlined piers were parametrized using Bézier cubic curves and an optimum cross section was determined in terms of maximum shear stress value. In a follow-up study, Tao and Li [24] attempted to investigate the effect of vertical profile curvatures. Similar to the previous study, vertical profile of the bridge piers were parametrized using Bézier cubic curves and optimum pier shape was obtained via numerical modelling. Differently, the reference cases in this study strictly prohibited downscaling of the cross sections to maintain the structural load bearing capacity. Implemented modifications were analyzed and the results suggested under this constraint the use of non-load bearing curtain walls. The main reason for preferring a non-curvilinear right triangular form is to weaken the impact of downward water jet and to guide the flow to adjust flow separation zone via an easy and practical shape to construct. Previous studies revealed that the sloped walls in pier geometry play also an important role in terms of flow forces acting on streambed [24, 25]. Throughout this study, corresponding terms to describe modified piers are classified and will be referred as illustrated in Figure 1 and summarized below (Table 1).

Table 1. Summary of the modified pier case classifications. “d” is the diameter of the pier

		H				
	d x	**1.0**	**0.8**	**0.6**	**0.4**	**0.2**
B	**0.1**	H1.0B0.1	H0.8B0.1	H0.6B0.1	H0.4B0.1	H0.2B0.1
	0.2	H1.0B0.2	H0.8B0.2	H0.6B0.2	H0.4B0.2	H0.2B0.2
	0.3	H1.0B0.3	H0.8B0.3	H0.6B0.3	H0.4B0.3	H0.2B0.3
	0.4	H1.0B0.4	H0.8B0.4	H0.6B0.4	H0.4B0.4	H0.2B0.4
	0.5	H1.0B0.5	H0.8B0.5	H0.6B0.5	H0.4B0.5	H0.2B0.5

ANSYS Fluent solver was used in this study. A total of 50 numerical runs, except for the reference cases were performed, 25 for each circular and oblong pier under same flow conditions. Analyses were repeated for different width (b) and height (h) combinations of the implemented triangle. Dimensional details of the solution domain and boundary conditions are shown in Figure 2.

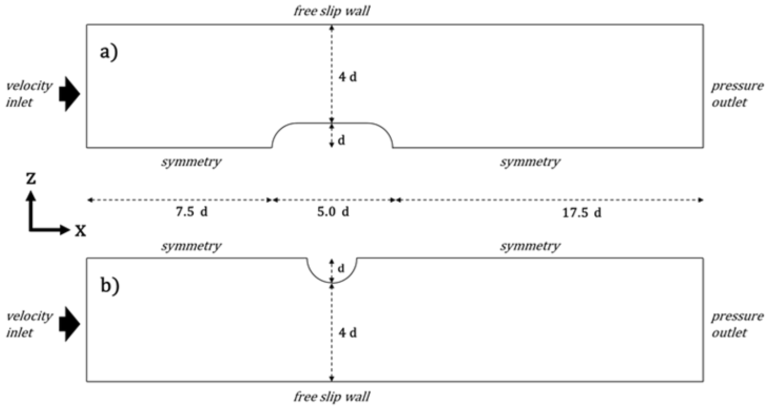
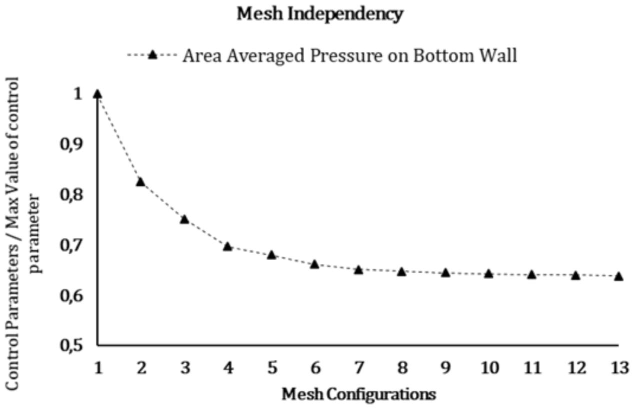


Figure 2. Dimensions of the solution domain and boundary conditions (figure is not scaled) a) oblong case b) circular case

Center of both oblong and circular piers were located 10d far from the inlet, 20 d from the outlet and 5d from the wall in span wise direction to eliminate the influence of boundaries on solutions. Velocity inlet and atmospheric pressure outlet condition applied. For low Froude numbers as in the current cases, effects of free surface profile can be negligible [26]. Accordingly, top of the domain was defined as a free slip wall to represent free surface as a rigid lid approximation. Far span-wise walls of the domain were also defined as free slip wall while pier and bottom walls were set as no-slip walls. Symmetry boundary was preferred along the centerline of the domain (Fig. 2).

Size and the shape of the mesh elements play an important role in the accuracy of the results. Mesh elements should be fine enough to capture interested flow scale and need to meet certain shape and size quality requirements while considering computational cost. For this reason, mesh size and configuration were subjected to a mesh independency test. Area averaged pressure on the riverbed was selected as the control parameter, characterizing the flow in the domain. Successive solutions from coarse to fine mesh elements were conducted until the variations in the control parameter was insignificant (Figure 3).



	1	2	3	4	5	6	7	8	9	10	11	12	13
Total Mesh Elements	864	2916	6912	13500	23328	37044	55296	78732	108000	143748	186624	237276	296352

Figure 3. Mesh independency test for different mesh configurations

As shown in Figure 3, a total of 13 mesh configurations were tested and configuration #11 with 186624 mesh elements was selected. Generated grid consists of pure hexahedral constructed mesh elements and it is refined close to the pier and bottom walls of the domain with a smooth transition between layers by adjusting aspect ratios of the elements (Figure 4).

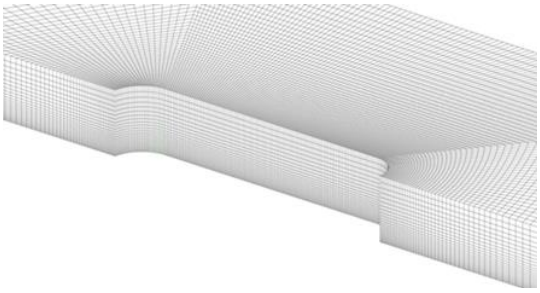


Figure 4. Isometric view of selected mesh configuration.

All runs were conducted under same initial conditions, flow velocity and water depth in the domain. Turbulent flow with a Reynolds number of approximately 200000 was modeled using standard k-ε turbulence closure. The k-ε is a two-equation turbulence model that is frequently used in both industrial and academic applications. This model is preferred because it requires lower computational load compared to other models which is an important factor for parametric CFD studies including numerous simulations. Scalable wall function was used to force y+ value to stay at the intersection of laminar sub-viscous layer (y+ > 11.225) and log law layer for varying adjacent wall distances and flow velocities. All steady-state iterative solutions were carried out until the same control parameters in mesh independency test reached to a steady form. Coupled algorithm for pressure-velocity coupling and second order schemes were used for numeric solutions of pressure, momentum and turbulence to improve accuracy.

3. Results

Performance of the modified piers were compared in terms of pressure and the wall shear stresses acting on riverbed as the two important flow forces that are responsible for triggering and developing scour. The effect of the inclined wall in reducing downward water jet induced high pressure on the bottom wall at the nose of piers were accordingly investigated. Figure 5 a) and b) present the maximum normalized pressure values (max pressure value for each case/ max value of all max pressure in design group) obtained from the successive analysis of all pier modifications for both circular and oblong cases. Trend of maximum pressure values of both two cases look very similar visually except some minor differences. For all cases, maximum pressure value was observed to be decreasing with the increase in B value as expected. These results confirm that the inclined wall with high aspect ratios leads to more reduction on maximum pressure values. In addition to the pressure, the wall shear stress on bottom wall is considered as another key factor. The maximum shear stress values for both circular and oblong cases are shown in Figures 6 a) and b). Similar to the pressure, maximum shear stress is decreasing with increase in B values which reaches to significantly low ratios for low H values. When compared with pressure, B value, particularly at its lower range, is less effective in reduction of wall shear stress. That is, reduction in shear stress is about 5% in case B0.1 where as it is 10% pressure reduction for different H values.

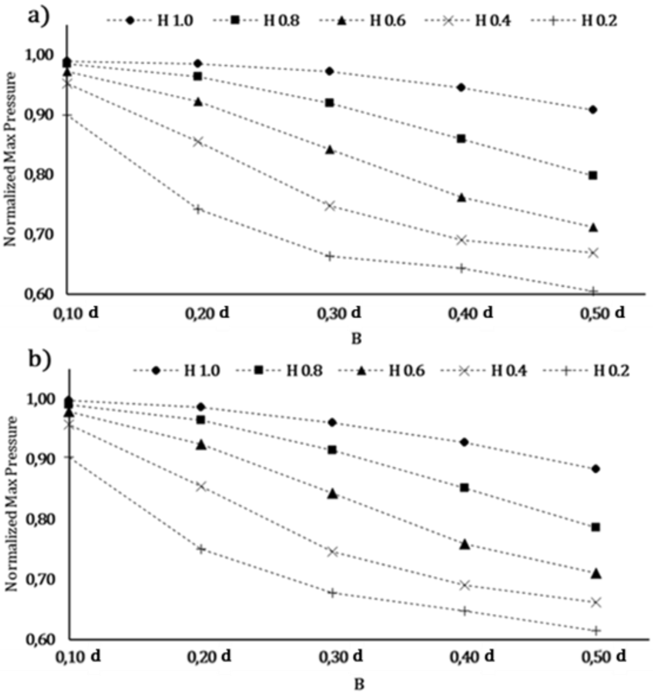


Figure 5. Normalized maximum pressure on riverbed (max pressure value for each case/ max value of all max pressure in design group) for different B/H
a) circular case b) oblong case

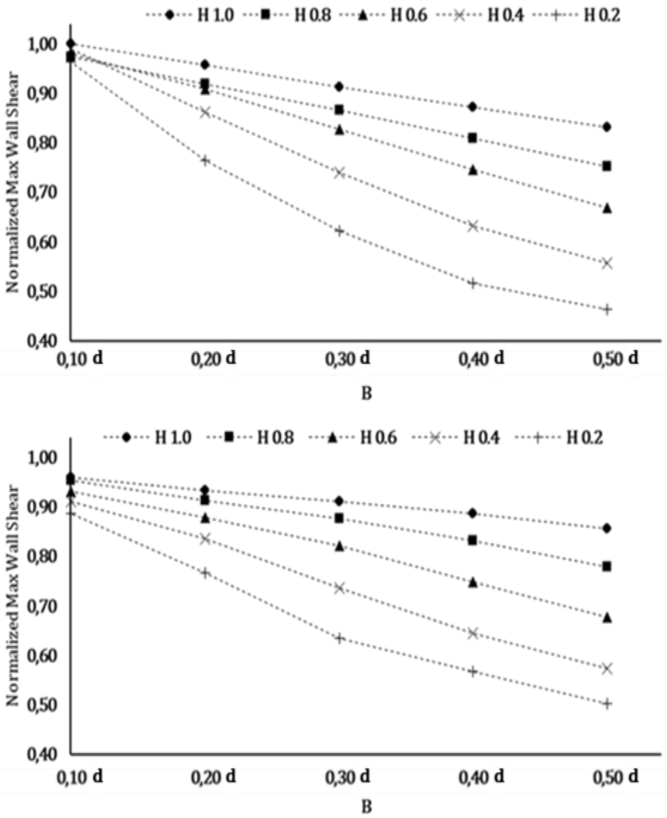


Figure 6. Normalized maximum wall shear on riverbed (max wall shear value for each case/ max value of all max wall shear in design group) for different B/H
a) circular case b) oblong case

While the maximum pressure and shear stress values are important scouring initiators, an assessment based on these parameters without considering the surface area where the forces act on cannot be complete. Figure 7 shows the spatial distribution of normalized shear stress values (max wall shear value for each case/ max value of all max wall shear in design group) that are greater than 0.5 near the pier. For almost all H values, maximum shear stress values decrease while the influence area increases. This result supports the requirements of an area weighted approach for both pressure and shear stress values to elaborate on the performance of modified piers. To obtain normalized values, the area averaged normalized pressure and shear stress values multiplied with the areas where the normalized pressure and shear stress values are greater than 0.5. The calculated force values are shown in Figure 8.

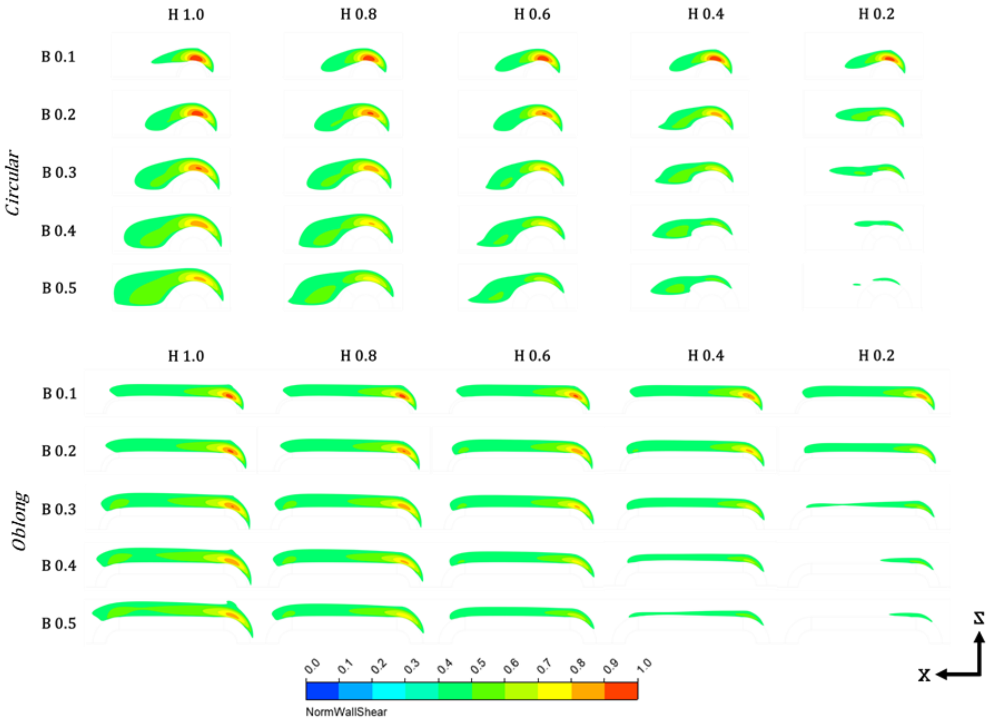


Figure 7. Distribution of normalized bed shear stress (max wall shear value for each case/ max value of all max wall shear in design group) > 0.5

From Figures 8 and 9 it can be argued that the most favorable case is still H0.2B0.5 in terms of normalized pressure and shear forces. What is interesting about the data in these figures is that all H series lead to increase in force with the increase in B except H0.2 case. Apparently, high aspect ratios (B/H) for implemented inclined walls does not perform as expected which can be explained by the increase in size of pier which corresponds higher flow resistance and higher surface area for stagnation pressure. Another explanation for this can be attributed to higher flow separation in gradually changed cross section than the sudden change in lower H values. High velocity flow that hit the smaller surface area at the higher H values causes lower downward flow force. Then its energy is dispersed by the highly skewed inclined wall with low H and high B values. All together, these results suggest that there is a tradeoff between maximum stress and area averaged forces to prevent initiation and development of scour.

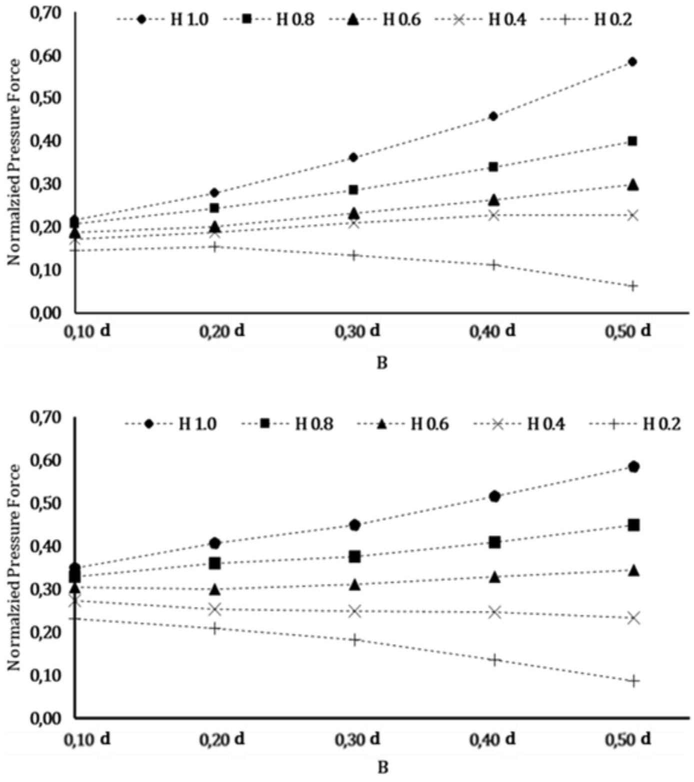


Figure 8. Normalized pressure force on riverbed for different B/H
a) circular case b) oblong case

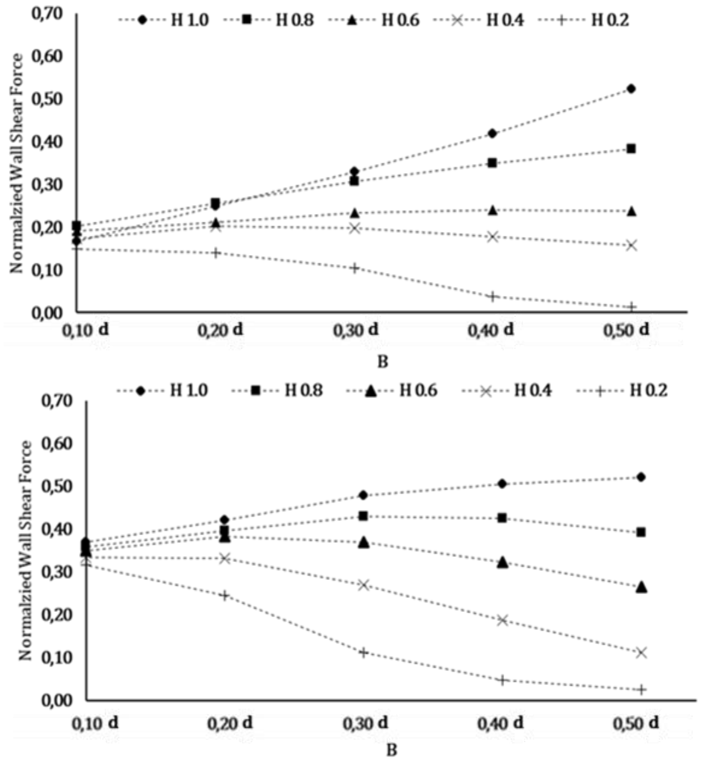


Figure 9. Normalized wall shear force on riverbed for different B/H
a) circular case b) oblong case

As previously stated, the flow around the piers generate complex vortex structures which is directly related to scour mechanism. Many researchers [27, 28] highlight the LES (Large Eddy Simulation) method that is more suited for resolving unsteady complex vortical flows. LES is an accurate and valid approach for a highly detailed examination of a single case. On the other hand, in terms of computational cost, Reynolds-averaged N-S models still represent an efficient way of modelling parametrized 3D analysis which consists of numerous simulations. In figure 10, the iso-surfaces of vortices are shown using Q criterion for both reference cases and H0.2B0.5 cases. Particularly, legs of the necklace vortex in reference cases extends longer distances than the modified ones. In addition to this, position of the vortex at the wake region of piers were shifted to an upper location in modified pier cases. This is presumably due to the fact that the flow guided by inclined wall interacts with the approaching flow at a higher elevation to generate wake vortex. Apparently, observed improvement in both necklace and wake vortices seem to be consistent with the shear stress results.

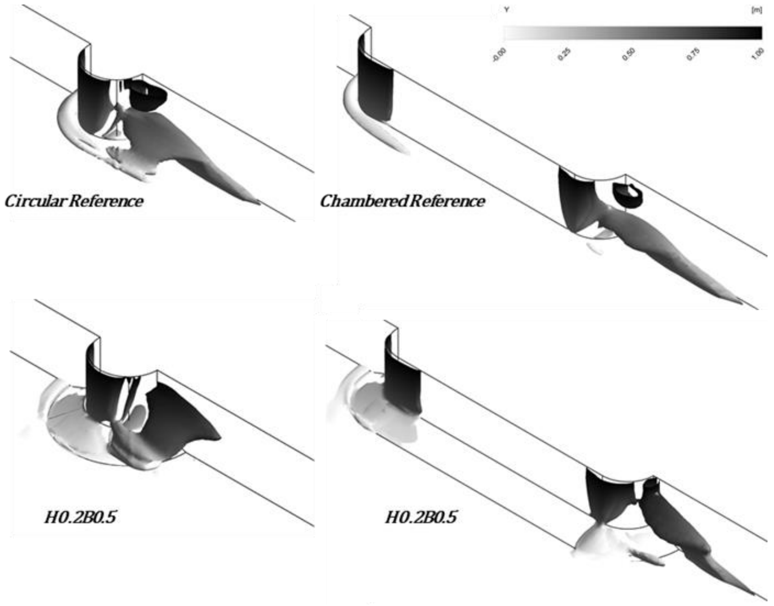


Figure 10. Vortex structures visualized using Q criterion colored according to elevation.

To have a closer look in the flow development in stream-wise direction and examine the trailing circulation zone, velocity distributions with velocity vectors are presented below at three successive sections (Figure 11 and 12). The flow separation induced high velocity loses its influence for oblong case due to the excessive length at the “a” section. Towards downstream at sections “b” and “c” of the reference cases, two circulations zone can be seen, one close to the bottom wall that is rotating in the clockwise direction with a wide influence area and the other at a higher elevation that is rotating in the counter clockwise direction with a small influence area. In the modified pier, the circulation zone moves away from the riverbed while getting a wider shape that reduces the interaction between the circulation zone and the bed material.

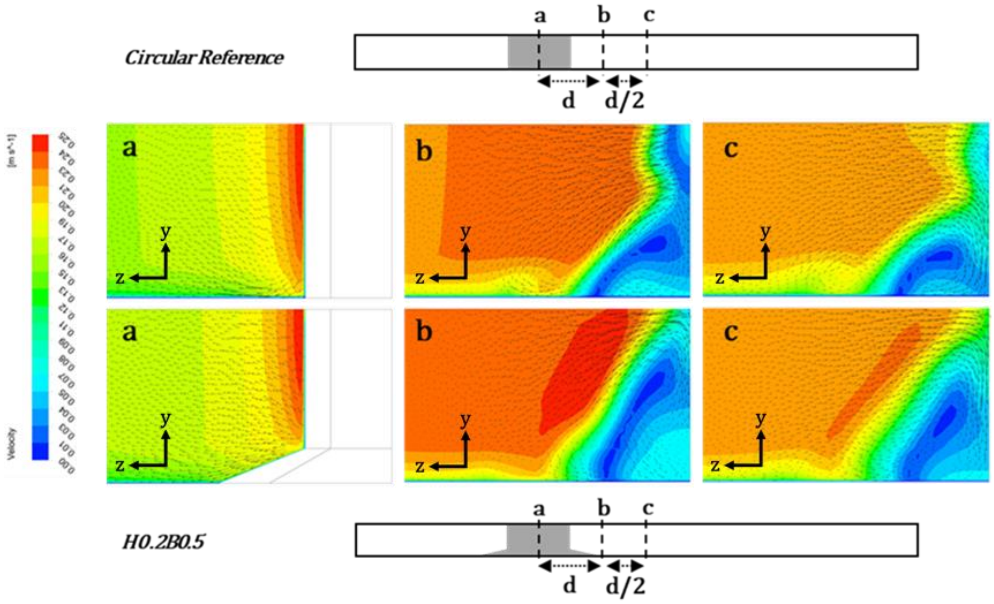


Figure 11. Velocity vector and distributions in stream-wise direction for the circular reference and H0.2B0.5 cases.

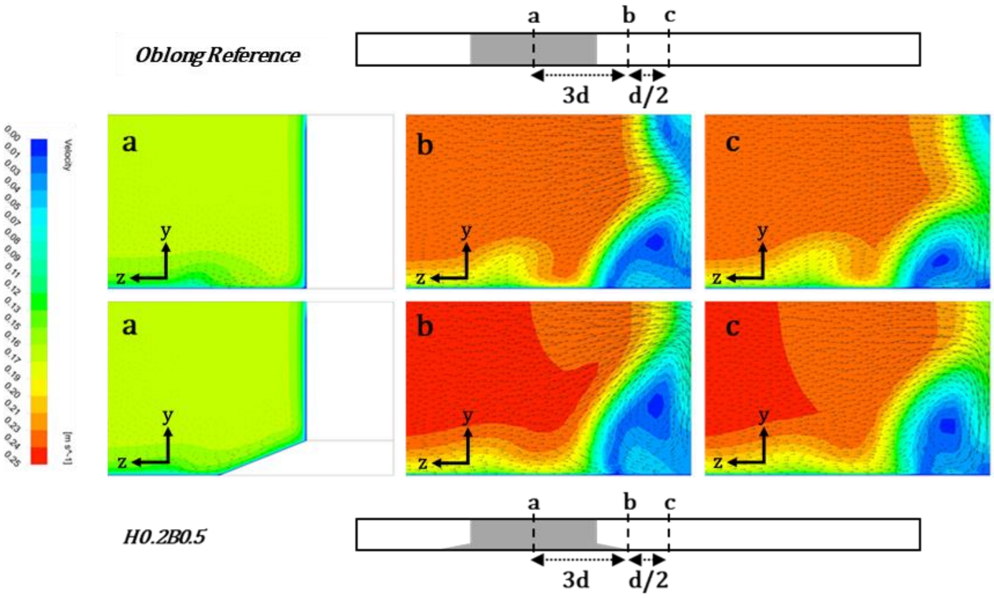


Figure 12. Velocity vector and distributions in stream-wise direction for the oblong reference and H0.2B0.5 cases.

4. Conclusion

The aim of the present study was to examine the flow forces acting on riverbed near bridge piers and present structural modifications to reduce local scour of two commonly used bridge piers using 3D CFD simulations. Cross sections of the pier geometries designed in this analysis used only extensions not reductions due to structural constrains. A right triangular implementation that wraps the reference shape was accordingly applied as non-load bearing curtain wall. Triangular implementation has been parametrized by its leg lengths (B and H) and successive numerical analyses were conducted. Pressure and wall shear stress acting on riverbed used to assess the performance of each modified case.

The presented modifications were designed to reduce the impact of downward water jet and to modify the flow separation zone by the gradually sloped wall of cone-like piers. Maximum pressure at the nose of piers and the maximum value of the shear stress around the piers near the bed and the vortex structures around the piers were the scour indicating parameters. The results show that low H values with high B values perform better in terms of reducing maximum pressure and wall shear. Reduction in both maximum pressure and wall shear stress reach up to 50% compared to the reference case. Pressure and shear stress values were also considered with their influence areas together to reflect general characteristics. That is, area averaged pressure and shear stress values were multiplied with the influence area to obtain a force parameter. Differently from the max values, results of area averaged approach indicate that the higher B values do not perform well for all H series except H0.2 case while the most favorable case in both analyses is H0.2-B0.5. In addition to these findings, vortex structures were also examined. Flow field for the H0.2-B0.5 case was compared with the reference case and it was observed that the position and the shape of the circulation zones are in a well agreement with the previous findings. In both circular and oblong cases, necklace vortex is shortened and downstream trailer circulation zone shifted to higher positions due to the applied geometric modifications.

All together, these findings suggest that a basic and practical structural modification has the potential to reduce the local scour near existing bridge piers. Multiple CFD analysis of parametrized geometric changes revealed the better option among different modifications. The study also contributes to existing knowledge and future efforts for an optimum bridge pier shape against local scour problem. It should be noted that the results presented here using k-e turbulence model can prove an effective way of guiding the design process by relative comparison of the performances of the new structures in the design pool. Final design still requires different and more complex modeling approaches such as LES and physical experiments for validation.

References

[1] Shirole, A.M. and Holt, R.C., Planning for a comprehensive bridge safety assurance program, **Transportation Research Record**, 1290, 39-50, (1991)

[2] Briaud, J.L., Ting, F.C., Chen, H.C., Gudavalli, R., Perugu, S. and Wei, G., SRICOS: Prediction of scour rate in cohesive soils at bridge piers, **Journal of Geotechnical and Geoenvironmental Engineering**, 125, 4, 237-246, (1999).

[3] Wardhana, K. and Hadipriono, F.C., Analysis of recent bridge failures in the United States, **Journal of Performance of Constructed Facilities**, 17, 3, 144-150, (2003).

[4] Melville, B.W. and Sutherland, A.J., Design Method for Local Scour at Bridge Piers, **Journal of Hydraulic Engineering**, 114, 10, 1210–1226, (1988).

[5] Melville, B.W. and Chiew, Y., Time Scale for Local Scour at Bridge Piers, **Journal of Hydraulic Engineering**, 125, 1, 59–65, (1999).

[6] Yanmaz, A.M. and Altinbilek, H.D., Study of Time-Depenbent Local Scour around Bridge Piers, **Journal of Hydraulic Engineering**, 117, 10, 1247–1268, (1991).

[7] Yanmaz M.A., Uncertainty of Local Scouring Parameters Around Bridge Piers, **Turkish Journal of Engineering and Environmental Science**. 25, 127-137, (2001).

[8] Kothyari U.C., Garde R.J. and Ranga R.K.G., Temporal variation of scour around cylindrical bridge piers, **Journal of Hydraulic Engineering**, 118, 8, 1091−1106, (1992).

[9] Kandasamy J.K. and Melville B.W., Maximum local scour depth at bridge piers and abutments, **Journal of Hydraulic Research**, 36, 183−197, (1998).

[10] Richardson, J.E. and Panchang, V.G., Three-Dimensional Simulation of Scour-Inducing Flow at Bridge Piers, **Journal of Hydraulic Engineering**, 124, 5, 530–540, (1998).

[11] Salaheldin, T.M., Jasim, I. and Mohammad, H.C., Numerical Modeling of Three-Dimensional Flow Field Around Circular Piers, **Journal of Hydraulic Engineering**, 130:91-100, (2004).

[12] Zhu Z.and Liu, Z., CFD prediction of local scour hole around bridge piers, **Journal of Central South University**, 19, 273−281, (2012).

[13] Xiong, W., Cai, C.S., Kong, B. and Kong, X., CFD simulations and analyses for bridge-scour development using a dynamic-mesh updating technique, **Journal of Computing in Civil Engineering**, 30, 1, 04014121, (2016).

[14] Nasr-Allah, T.H., Moussa, Y.A.M, Abdel-Aal, G.M. and Awad, A.S., Experimental and numerical simulation of scour at bridge abutment provided with different arrangements of collars, **Alexandria Engineering Journal**, 55, 1455–1463, (2016).

[15] Li, J. and Tao, J., CFD-DEM Two-Way Coupled Numerical Simulation of Bridge Local Scour Behavior under Clear-Water Conditions, **Transportation Research Record**, 2672, 39, 107-117, (2018).

[16] Chiew, Y., Scour Protection at Bridge Piers, **Journal of Hydraulic Engineering**, 118, 9, 1260–1269, (1992).

[17] Kumar, V., Kittur, G.R.R. and Nandana, V., Reduction of local scour around bridge piers using slots and collars, **Journal of Hydraulic Engineering**, 125, 1302-1305, (1999).

[18] Zarrati, A.R., Gholami, H. and Mashahir, M.B., Application of collar to control scouring around rectangular bridge piers, **Journal of Hydraulic Research**, 42, 1, 97-103, (2004).

[19] Fotherby, L.M., Alternatives to riprap for protection against local scour at bridge piers, **Transportation Research Record**, 1420: 32-39, (1993).

[20] Pasha, M., Mahmood, A.H. and Shams, S., An Analysis of Scouring Effects on Various Shaped Bridge Piers, **Brunei Darussalam Journal of Technology and Commerce**, 7, 29-42, (2013).

[21] Vijayasree, B.A., Eldho, T.I., Mazumder, B.S.and Ahmad, N., Influence of bridge pier shape on flow field and scour geometry, **International Journal of River Basin Management**, 17, 1, 109-129, (2019).

[22] Yagci, O., Yildirim, I., Celik, M.F., Kitsikoudis, V., Duran, Z. and Kirca, V.S.O., Clear water scour around a finite array of cylinders, **Applied Ocean Research.**, 68, 114–129, (2017).

[23] Li, J. and Tao, J., Streamlining of bridge piers as scour countermeasures: optimization of cross sections, **Transportation Research Record**, 2521, 1, 162-171, (2015).

[24] Li, J., Tao, J. and Yu, X., Streamlining of Bridge Pier as a Scour Countermeasure: A Feasibility Study, Proc. **International Foundations Congress and Equipment Exposition**, 319–329, (2015).

[25] Koken, M. and Constantinescu, G., Flow and Turbulence Structure Around Abutments with Sloped Sidewalls, **Journal of Hydraulic Engineering**, 140, 7,

04014031, (2014).

[26] Roulund, A., Sumer, B. M., Fredsoe, J., and Michelson, J., Numerical and experimental investigation of flow and scour around a circular pile, J**ournal of Fluid Mechanics**, 534, 351–401, (2005).

[27] Simpson, R.L., Junction flows, **Annual Review of Fluid Mechanic**s, 33, 1, 415-443, (2001).

[28] Chang, W.Y., Constantinescu, G., Lien, H.C., Tsai, W.F., Lai, J.S. and Loh, C.H., Flow structure around bridge piers of varying geometrical complexity, **Journal of Hydraulic Engineering**, 139, 8, 812-826, (2013).